Partitioned Fluid-Structure Interaction with Fluent

Joris Degroote, Jan Vierendeels
Joris.Degroote@UGent.be, Jan.Vierendeels@UGent.be
Ghent University, Department of Flow, Heat and Combustion Mechanics,
Sint-Pietersnieuwstraat 41, B-9000 Ghent, Belgium

The simulation of fluid-structure interaction (FSI) and multi-physics in general has gained in importance in both academia and industry. In an FSI simulation, the fluid domain and the structural domain do not overlap and they only interact through their common boundary: the flow exerts a force on the structure and the displacement of the structure influences the flow.

It is possible to simulate this interaction with a single code for the flow and the structure, the so-called monolithic approach [2]. However, a new code has to be developed for this specific combination of physical problems. The monolithic approach does not take advantage of the mature computational fluid dynamics (CFD) and computational structural dynamics (CSD) codes that are readily available. Therefore, FSI research at Ghent University is focused on the partitioned approach which means that existing CFD and CSD codes are coupled.

Special coupling techniques have been developed to couple commercial codes like Fluent with CSD codes [1; 3]. These techniques treat the CFD and CSD code as a black box and can be used for both steady and unsteady simulations. Coupling iterations between the CFD and CSD code have to be performed in an unsteady simulation with an incompressible fluid. The coupling algorithm prescribes the position of the fluid-structure interface and Fluent solves the flow problem using its dynamic mesh capability. With the resulting stress distribution on the fluid-structure interface, the coupling algorithm calculates the stress distribution that has to be applied on the boundary of the structure and the displacement of the structure is subsequently calculated with a finite element CSD code. The coupling algorithm then calculates the new position of the fluid-structure interface and iterations are performed until the solution has been found.

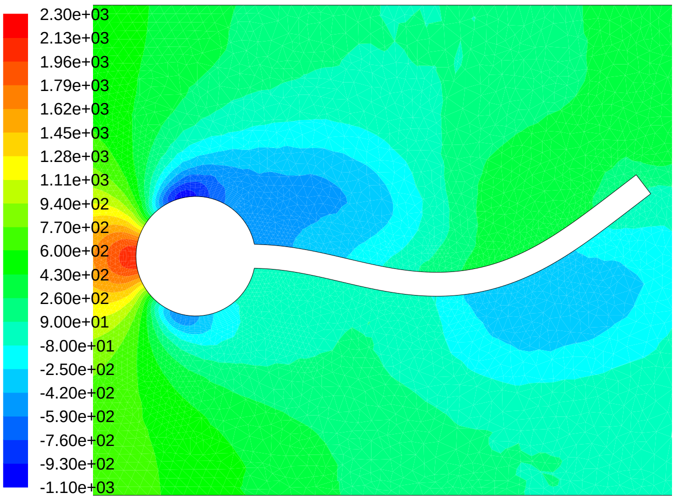
The FSI calculation has been accelerated by running Fluent in parallel and by using the Scheme interface and user-defined functions to solve the flow problem in consecutive coupling iterations starting from the solution of the previous coupling iteration. This has been used to solve FSI problems in 2D and 3D. A first example is the 2D benchmark of Turek et al. [2]. It consists of a flexible beam behind a cylinder in a laminar flow. The pressure contours in the unsteady FSI2 test after 12 and 16 s are shown in Figure 1. A second example is a 3D simulation of the propagation of a pressure pulse in an artery as shown in Figure 2.

Bibliography

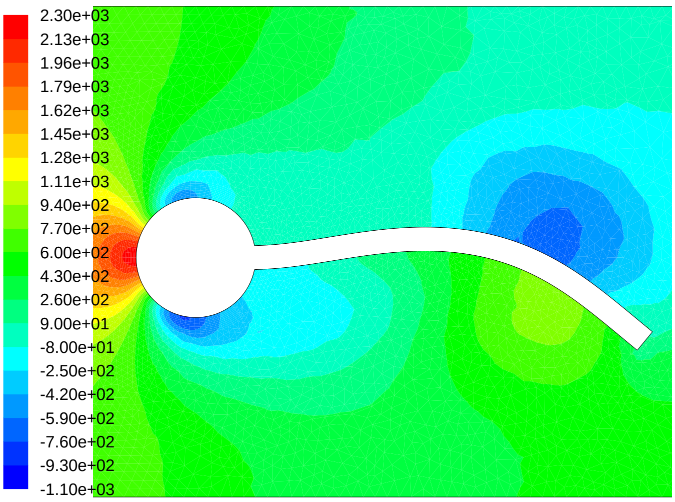
[1] J. Degroote, P. Bruggeman, R. Haelterman, and J. Vierendeels. Stability of a coupling technique for partitioned solvers in fsi applications. *Computers and Structures*, 0(0):0, 2008. In press, DOI: 10.1016/j.compstruc.2008.05.005.

[2] S. Turek and J. Hron. Proposal for numerical benchmarking of fluid-structure interaction between an elastic object and laminar incompressible flow. In H.-J. Bungartz and M. Schäfer, editors, *Fluid–Structure Interaction – Modelling, Simulation, Optimisation*, number 53 in Lecture Notes in Computational Science and Engineering, pages 371–385. Springer, Berlin, 2006. ISBN 3-540-34595-7.

[3] J. Vierendeels, L. Lanoye, J. Degroote, and P. Verdonck. Implicit coupling of partitioned fluid-structure interaction problems with reduced order models. *Computers and Structures*, 85(11–14):970–976, 2007.

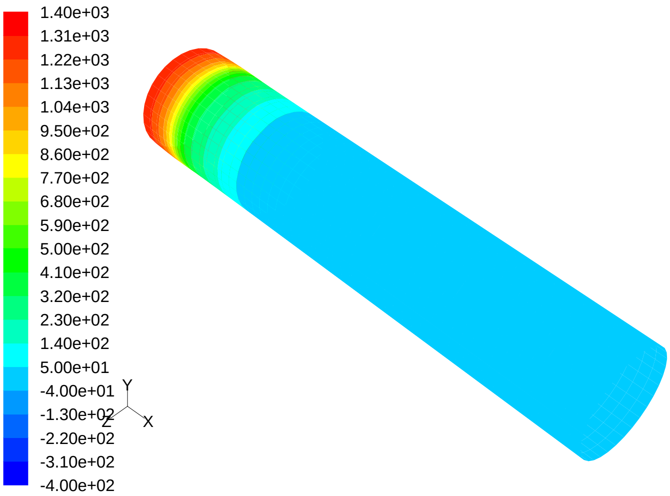


(a)

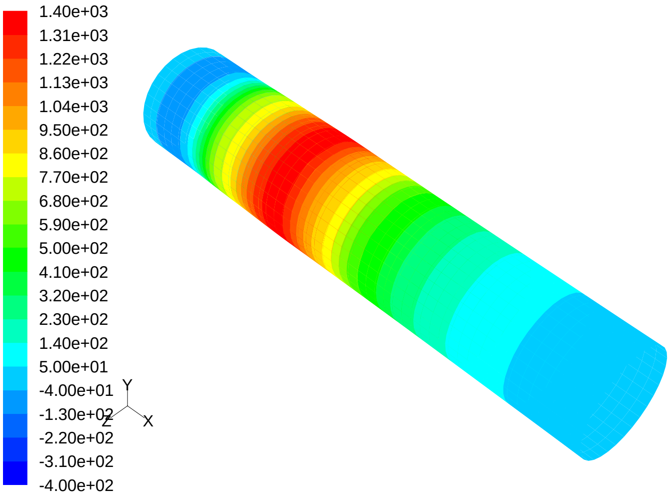


(b)

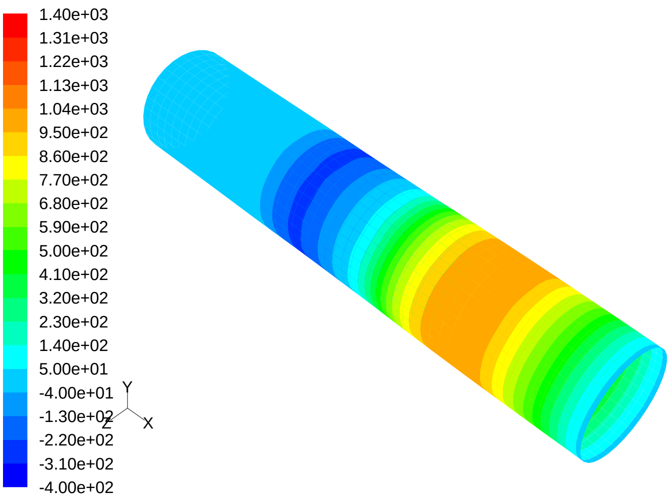
Figure 1: Pressure contours in the 2D unsteady FSI2 test with the flexible beam after (a) 12 s; (b) 16 s.



(a)



(b)



(c)

Figure 2: Pressure contours on the fluid-structure interface in a 3D simulation of the flexible artery after (a) 10^{-3} s; (b) $5{\cdot}10^{-3}$ s; (c) $9{\cdot}10^{-3}$ s.